archives
of thermodynamics
Vol. **32**(2011), No. 3, 263–272
DOI: 10.2478/v10173-011-0027-z

Optimization of guide vane positions in bended inflow of mechanical draft wet-cooling tower

ADAM KLIMANEK[a*]
TOMASZ MUSIOŁ[a]
ADAM STECHMAN[b]

[a] Silesian University of Technology, Institute of Thermal Technology, ul. Konarskiego 22, 44-100 Gliwice, Poland

[b] Projchłod Sp. z o.o., ul. Góry Chełmskiej 2B, 44-100 Gliwice, Poland

Abstract Optimization of vane positions in a mechanical draft wet-cooling tower is presented in this paper. The originally installed, equally spaced, vanes produced non-uniform air velocity distribution reducing the performance of the fill of the cooling tower. A 2D CFD model of the tower has been created. The model has then been used to determine the objective function in the optimization procedure. The selected objective function was the standard deviation of the velocity of air entering the fill. The Goal Driven Optimization tools of the ANSYS Workbench 2.0 have been used for the optimization and the ANSYS Fluent 13.0 as a flow solver. The optimization allowed reduction of the objective function and producing a more uniform air flow.

Keywords: Optimization; mechanical draft wet-cooling towers; Multiphase flow

1 Introduction

Cooling towers are devices used to cool industrial water, where the waste heat is transferred to the air stream. The frequently encountered cooling

*Corresponding author. E-mail address: adam.klimanek@polsl.pl

towers of large industrial systems are the natural draft ones, where the flow of air is induced by the difference in densities of the air inside the tower and in the atmosphere. For smaller systems usually mechanical draft cooling towers are used where the flow of cooling air is forced by a fan. For medium size plants both natural and mechanical draft towers are used with the latter being arranged in-line to form banks of individual cells. The reasons for that are the sizes of the mechanical draft towers, which are limited by the fan diameter. The mechanical draft towers usually allow obtaining lower cooled water temperatures than the natural draft towers. The main advantages of mechanical draft towers are the low capital and construction cost, high air flow at all loads and atmospheric conditions and small dimensions. The fans, however, require electrical energy supply that increases the running cost [1]. The mechanical draft towers are also used as test facilities of the cooling tower equipment since the flow of cooling air can be controlled by the fan speed.

In wet or evaporative cooling towers the flowing air is in direct contact with the water to be cooled. The reciprocal air and water flow directions may be different depending on the arrangement of the fill. As presented in Fig. 1, counterflow and crossflow arrangements can be distinguished. In the counterflow systems the water flows downwards and the air upwards. The hot water is sprinkled on the heat and mass exchanger called a fill whose aim is to extend the heat and mass transfer area between water and air. After leaving the fill the water falls through the so called rain zone to the water basin, where it is collected and pumped back to the system.

Turning or guide vanes are often used in ducts to direct flowing gases around bends of the ducts. The main reason for their use is the pressure loss due to eddy zone formed near the inner wall of the duct. Appropriately selected vanes can diminish or eliminate the eddy zone and reduce this pressure loss considerably [2]. The guide vanes are also applied in mechanical draft cooling towers to reduce the losses as well as direct and uniformly distribute the flow of air under the tower. The uniformity of the air flow affects the overall efficiency of the system. On one hand the increase of air velocity increases the heat and mass transfer reducing the cooled water temperature but on the other increases the pressure drop in the system. Therefore there is a need for uniform distribution of the air flow through the fill.

In cooling towers the turning vanes or louvers are usually installed uniformly. Their positions and orientations are rarely optimized. Practical

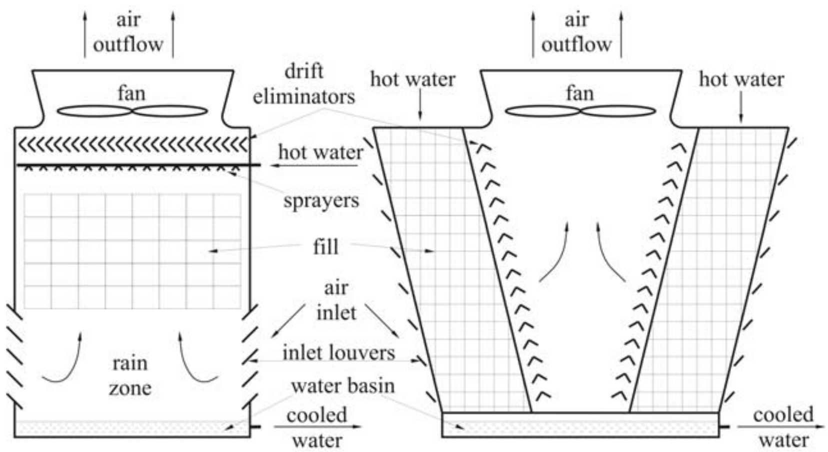

Figure 1. Mechanical draft cooling towers with counterflow (left) and crossflow (right) fill arrangements, according to [1].

recommendations for vanes selection based on experimental studies can be found in [2]. In this paper optimization of the vanes positions installed in an experimental mechanical draft cooling tower is studied. The ANSYS Fluent 13 code is used to simulate the flow of air and water through the tower. The objective function used is the standard deviation of the air velocity at the bottom of the fill. The Goal Driven Optimization tool of ANSYS Workbench 2.0 has been used for the optimization. It should be stressed however that it is not known whether the uniform air flow produces a global optimum performance of the tower. The reason for that is usually non-uniform water flow through the tower and the contrary acting effects of heat and mass transfer and pressure drop.

2 The studied tower

The mechanical draft cooling tower under consideration is an existing experimental facility built by Projchłod at Łaziska Power Plant in Poland. The schematic diagram of the tower is shown in Fig. 2. The fan (that is not shown) is placed at the top of the tower. The water is distributed above the fill zone and flows through the fill, rain zone and falls to the water basin at the bottom. The air is delivered through a 11.7 m long circular duct, whose end is shown in Fig. 2. The guide vanes are installed diagonally in the rain zone. There are 9 uniformly distributed vanes of the shape and dimensions presented in Fig. 3. The distance between each vane is 0.2882 m. The distance of the first vane from the wall is 0.2383 m. The thickness of each vane is 1 mm. The main pressure drop occurs in the fill. The pressure drop characteristics has been measured and presented by Stechman [3]. The used correlation for loss coefficient is given by

$$C_{fi} = 7.56 + 0.89 m_w \tag{1}$$

and the pressure drop per unit fill height is

$$S_{fi} = \frac{\Delta p}{H} = C_{fi} \frac{\rho u^2}{2} \;, \tag{2}$$

where m_w is the water mass flux (kg/m^{2}s), H (m) is the fill height, ρ (kg/m^3) and u (m/s)are air density and velocity, respectively. The pressure drop in the fill was taken into account by introducing sources to the momentum equations in ANSYS Fluent using the built in porous media model.

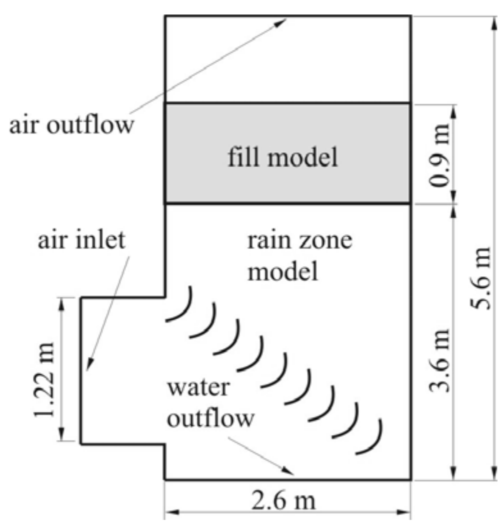

Figure 2. Schematic diagram of a mechanical draft cooling tower.

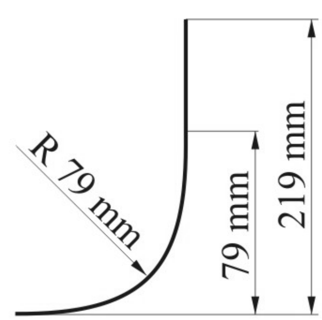

Figure 3. Shape and dimensions of a single guide vane.

3 Numerical model of the cooling tower

The numerical model has been developed using the ANSYS software. The geometry has been created in the Design Modeler. The geometry has been represented by a two dimensional model. The positions of the vanes were parameterized allowing. The computational mesh of ca. 35 000 grid cells was generated using the Meshing. The flow was simulated using the Fluent software. The solved equations were the continuity and momentum equations and two additional transport equations of the k-ε turbulence model. Both standard and realizable k-ε [4] models have been tested. The flow of the water droplets in the rain zone has been modeled using the discrete phase model [4], which tracks the droplets trajectories in a Lagrangian frame. The water droplets were assumed to have diameters of 5 mm. Kröger in [1] states that Sauter mean droplet diameters typically found under film fills are between 5 and 6 mm. The flow of water in the fill zone has not been modeled explicitly however the effects have been accounted for by applying Eq. (1). This is the model of the fill presented in Fig. 2.

3.1 Boundary conditions

Velocity inlet boundary condition has been used for the inflowing air and static pressure is specified at the top of the tower. To make the model simpler the flow of air in the air duct has been modeled separately. The resultant velocity profile has then been used as the boundary condition for the tower. The final length of the duct modeled is 1 m. The water droplets are introduced under the fill and escape the domain when they reach the bottom boundary. The no slip boundary condition has been specified on the tower walls and vanes. The wall function has been used to model the turbulent boundary layer on the walls and vanes. The water mass flux has been selected to be 3 kg/m^{2}s.

4 Optimization

Optimization of the vanes positions is performed using the design explorer of the ANSYS Workbench. The optimization is realized in a three step procedure in which first the design of experiments (DOE) is evaluated. For each sampling point the objective function is evaluated thus a computational fluid dynamics (CFD) simulation of the cooling tower is performed. In the second step, based on the generated values of the objective function in sampling points the response surface (RS) is generated. The last step is the optimization, which is performed on the Response Surface of the model. The optimum found on the Surface is then validated by performing a test simulation using the CFD model of the cooling tower. The workflow of the optimization procedure described above is presented in Fig. 4. The Central Composite Design has been used as the DOE type. It has however been modified to account for the constraints put on the parameters, as will be discussed later. The non-parametric regression has been used for the RS generation and Non-linear Programming by Quadratic Lagrangian (NLPQL) method has been utilized as the optimization algorithm. There are nine ($N = 9$) parameters (variables) x_i in the optimization procedure representing the distance of guide vane i to its neighbor on the left. The distance of the first vane is therefore to the wall of the cooling tower. The parameters were subject to the following constraints

$$0.1 \leq x_i \leq 2.0 \,, \tag{3}$$

$$\sum_{i=1}^{N} x_i \leq 2.688 \,. \tag{4}$$

Constraint (3) allows each vane to be at any position on the diagonal of the rain zone in the tower up to 2 m. Its position depends on the left neighbor. The constraint (4) does not allow the vanes to escape from the tower through the right wall. The optimization algorithm used (NLPQL) allows handling constraints (3) and (4). The DOE does not allow specifying constraint (4) since the design is generated for min and max values of the parameters x_i. This difficulty has been overcome by creating a custom Central Composite Design taking into account constraint (4). This has been achieved by secondary normalization of the design of sample points that did not obey (4). As mentioned before, the objective function was the standard deviation of air velocity at the inlet to the fill (cf. Fig. 2). The function is evaluated from the converged solution of flow in the cooling tower and can be written as

$$S = \sqrt{\frac{1}{n}\sum_{i=1}^{n}(u_i - \bar{u})^2} \rightarrow \min \,, \tag{5}$$

where n is the number of mesh faces on the inlet, u_i is the velocity magnitude at face i and $\bar{u}$ is the mean velocity given by

$$\bar{u} = \frac{1}{n}\sum_{i=1}^{n} u_i \; . \tag{6}$$

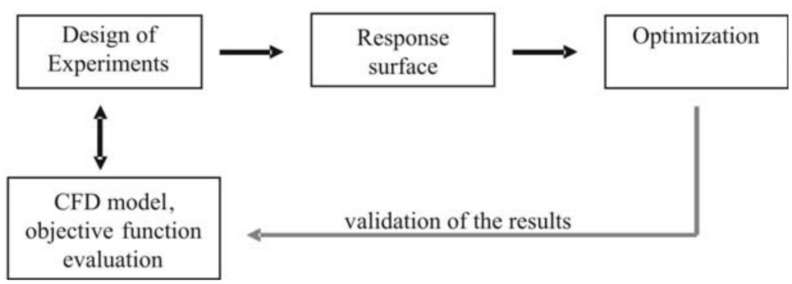

Figure 4. The workflow of the optimization procedure.

5 Results

The standard [6] and realizable [7] turbulence models have been tested in the course of the study. The effect of inclusion of the multiphase flow of droplets in the rain zone has also been examined. The model has also been tested for the independence of the solution of the mesh. Comparison of the obtained results is shown in Fig. 5. As can be seen, both the turbulence model and inclusion of the multiphase model in the rain zone have a significant effect on the obtained results. Finally the realizable k-ε has been selected and the flow of water droplets has been included in all further calculations. The case with these models included, obtained for the existing, uniform distribution of guide vanes is referred to as the reference case.

Relevant to the central composite design, 146 sample points have been generated for which objective functions have been determined using the CFD model. Based on these points a response surface has been calculated. A candidate point has been found during optimization on the response surface, which was then validated during a CFD simulation. The optimum solution for this point is termed *optimization 1.* The candidate point has also been used to refine the response surface in the vicinity of this optimum.

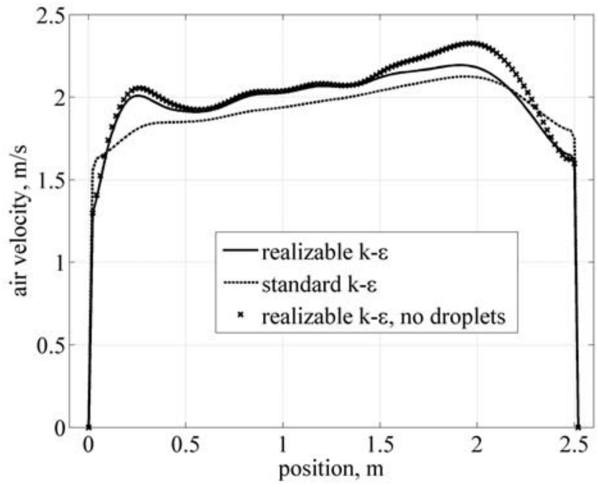

Figure 5. Profiles of velocity at the inlet to the fill for standard k-ε, realizable k-ε and realizable k-ε omitting the flow of water droplets in the rain zone.

A new candidate point termed *optimization 2* have been then found and validated. The profiles of velocity under the fill and their corresponding standard deviations obtained for the reference case and the two optimization results are presented in Fig. 6. As can be seen the differences in the obtained velocity profiles are not significant, however the optimization allowed reduction of the objective function by 22% for *optimization 1* and 45% for *optimization 2*. The obtained distributions of the guide vanes are presented in Tab. 1.

Table 1. Distributions of the guide vanes and their corresponding standard deviations.

Vane no.	1	2	3	4	5	6	7	8	9	S
reference	238.3	288.2	288.2	288.2	288.2	288.2	288.2	288.2	288.2	0.199
optim. 1	252.4	271.5	308.4	258.0	223.6	203.9	222.4	230.5	242.3	0.155
optim. 2	249.6	272.9	320.6	256.3	218.0	196.0	212.8	222.8	237.0	0.109

In Fig. 7 contours of velocity obtained for the reference and optimization 2 cases are presented. As can be seen the vanes in the optimized case are distributed closer to the center of the rain zone. This is justified since in the reference case the last three vanes lie in the zone of very small air flow. The distances between the vanes close to the wall are the largest.

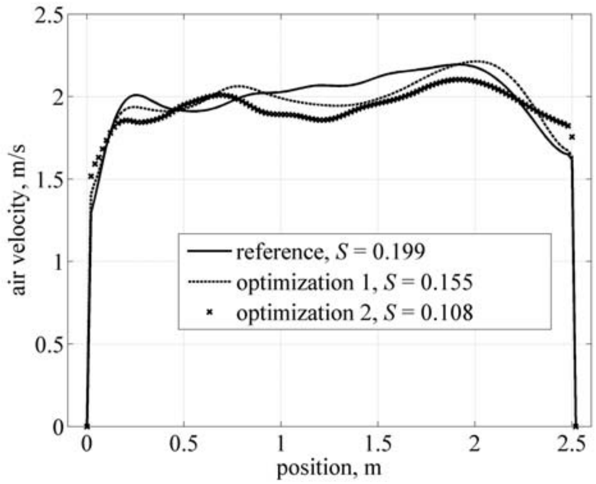

Figure 6. Profiles of velocity and their corresponding standard deviations of the reference and optimized vane distributions.

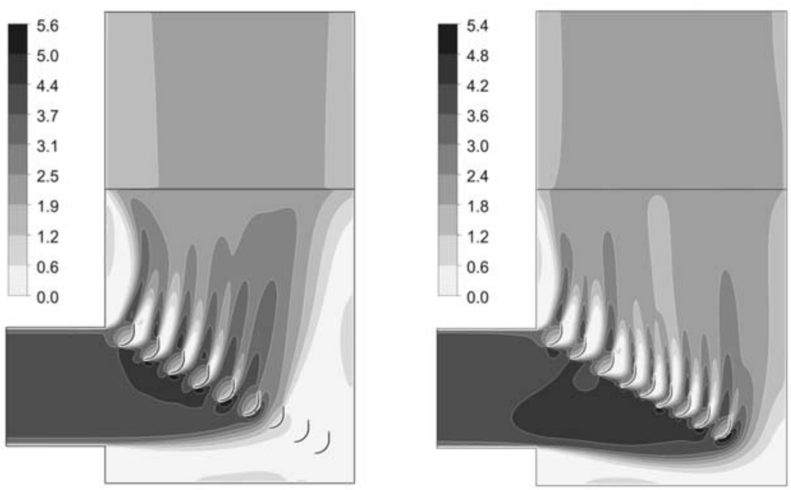

Figure 7. Contours of velocity in m/s for the reference (left) and optimized (right) vanes.

6 Conclusions

Optimization of guide vanes in a mechanical draft wet-cooling tower has been presented. The tools of the ANSYS 13.0 have been used in both the optimization as well as objective function evaluation for which a computational fluid dynamics code has been used. The CFD model has been tested by comparison of two turbulence models and mesh density. The effect of inclusion of the multiphase flow model of water droplets in the rain zone has been examined. Finally the model has been automated for the optimization. A three stage optimization procedure in which design of experiments, response surface and optimization on the surface were conducted. Two results have been obtained. The first result has been used as a refinement point for the response surface and second optimization. The obtained results showed that the optimal air velocity profiles do not differ significantly from the reference case; however the optimization allowed reduction of the objective function by 45%. The optimization showed that the guide vanes should be moved closer to the center of the rain zone for more uniformly distributed air flow. It should be stressed that it is not known if the uniform air flow produces a global optimum performance of the tower. The reason for that is usually non-uniform water flow through the tower and the contrary acting effects of heat and mass transfer and pressure drop. A more complex optimization procedure including the analysis of heat and mass transfer and pressure drop should be performed to address this problem.

Acknowledgements This work has been partially supported by RECENT 7FP project, grant no 245819.

Received 10 October 2011

References

[1] Kröger D.G.: *Air-Cooled Heat Exchangers and Cooling Towers*. Penn Well Corporation, Oklahoma 2004.

[2] Idelchik I.E.: *Handbook of Hydraulik Resistance*, 3rd edn., Begell House, 2001.

[3] Stechman A.: *Procedure for operation of natural draft wet-cooling towers in winter conditions*. PhD thesis, Silesian University of Technology, Gliwice 2006 (in Polish).

[4] *ANSYS FLUENT Theory Guide*, Release 13.0. ANSYS Inc. 2010.

[5] *ANSYS Help, Workbench*, Release 13.0. ANSYS Inc. 2010.

[6] Launder B.E., Spalding D.B.: *Lectures in Mathematical Models of Turbulence*. Academic Press, London 1972.

[7] Shih T.H., Liou W.W., Shabbir A., Yang Z., Zhu J.: *A new k-ε eddy viscosity model for high Reynolds number turbulent flows*; Computers & Fluids **24(1995)**, 3, 227-238.